Pertanika J. Sci. & Technol. 20 (2): 347 - 353 (2012)

SCIENCE & TECHNOLOGY

Journal homepage: http://www.pertanika.upm.edu.my/

Pressure Reduction on Blood Flow in Aorta Coronary Sinus Conduit

Siti Aslina Hussain[1*], Tan Hong Tat[1], Mohd Ismail Abdul Hamid[2], Norhafizah Abdullah[1] and Azni Idris[1]

[1]*Department of Chemical and Environmental Engineering, Faculty of Engineering, Universiti Putra Malaysia, 43400 Serdang, Selangor, Malaysia*

[2]*Middlesex Hospital, London*

ABSTRACT

Numerical studies of blood flow system of aorta coronary sinus conduit were carried out using ANSYS™ CFD simulation. A different model of conduit, which differs in the inlet diameter, was investigated. The investigated inlet diameters are 3 mm, 4 mm and 5 mm. Pressure drop from 80 mmHg to 15 mmHg was achieved for all the models. The comparison chart was produced to compare the pattern of pressure reduction as well as velocity distribution in each model. From the analysis of coronary sinus conduit, it was found that a narrow tube needs to be incorporated into the conduit produced. This is to induce a venturi effect to reduce the pressure drop of blood within a specific throat length. As conclusion, a model of 3 mm inlet and a throat diameter of 1.13 mm show satisfactory result for pressure reduction from 80 mmHg to 15 mmHg. This particular model also has a lower peak velocity at the inlet zone of the throat section, which is more preferable in terms of Reynolds number.

Keywords: Aorta coronary sinus conduit, blood flow, computational fluid dynamics, simulation, velocity

Article history:
Received: 12 April 2011
Accepted: 21 April 2011

Email addresses:
aslina@eng.upm.edu.my (Siti Aslina Hussain),
fizah@eng.upm.edu.my (Norhafizah Abdullah),
azni@eng.upm.edu.my (Azni Idris)
*Corresponding Author

INTRODUCTION

Fluid plays an important task in our daily life. The main fluid in human body is called blood. This particular fluid plays an important role in the human body as an oxygen delivery medium as well as a heat transfer agent. The oxygen that the blood carries is supplied to the muscles or cells in the body in order for it to function properly.

ISSN: 0128-7608

Each year, over a million people in the U.S. have a heart attack (World Health Organization, 2007). This symptom is also known as Acute Myocardial Infarction in the medical term. It is a medical condition that occurs when the blood supply to a part of the heart's muscles is interrupted. The result of ischemia or oxygen shortage will lead to damage and potential death of heart tissue. A classical treatment to heart attack is usually done by undergoing a bypass surgery, whereby the bypass grafts are frequently harvested from internal thoracic arteries, radial arteries or saphenous veins (Lee, 2006).

Nonetheless, the artificial conduits used in bypass surgery are subjected to analysis. The prime focus is particularly on the coronary sinus conduit. Coronary sinus is a collection of veins joined together to form a large vessel that collects blood from the myocardium of the heart. This sinus receives most of the venous blood from the heart and empties into the right atrium. It is located between the left atrium and ventricle on the posterior surface of the heart. It runs transversely in the groove between the left atrium and ventricle on the posterior surface of the heart (Syoten, 1980). An illustration of the exact location of the coronary sinus is shown in Fig.1.

A simulation was carried out in order to simulate the blood flow condition in the aorta coronary sinus. The main interest of the simulation was to observe the pressure distribution in the conduits, as well as the flow pattern. It is preferable to obtain a pressure drop from 80 mmHg at the inlet of the conduits to 15 mmHg at the outlet of the conduits. Besides that, the flow pattern must be laminar according to the specification.

MATERIALS AND METHODS

Computational fluid dynamics (CFD) technique was chosen in the simulation of blood flow. It is important to note that computational fluid dynamics is one of the branches of fluid mechanics

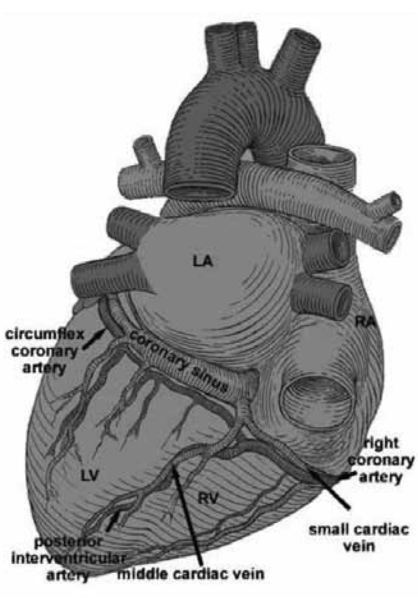

Fig.1: A posterior view of the heart

that uses numerical methods and algorithms to solve and analyze problems that involve fluid flows. Computers are used to perform the millions of calculations required to simulate the interaction of fluids with the complex surfaces used in engineering (Cengel & Cimbala, 2010).

The computational fluid dynamics software used in the simulation of blood flow in the aorta coronary sinus is ANSYS™ CFX. This software used finite element method in the analysis of the flow model. The delivery of the results of the analysis is by numerical method. The fundamental equation lies behind this software is the Navier-Stokes equations.

The purpose of using CFD to simulate the blood flow in the artificial coronary sinus model is because it helps to reduce the time and effort needed to run a real experiment which may not be cost effective. In addition, the actual size of the model will be too difficult to be analyzed directly using the present equipment. Moreover, certain part of the model is too narrow to be physically tested with the normal pressure devices and flow devices. In contrast, several variables and data can be extracted from the result using computational modelling. If there is any amendment to the physical model, the analysis process can be repeated quickly to reflect the result.

The project was carried out in 3 basic phases, namely pre-modelling, modelling and simulation. All the models produced will undergo these 3 phases and their outcome will be summarized. In the process of producing the simulation, the model will undergo each section of the programme in ANSYS™ CFD, such as Design Modeller, Meshing, Pre-Processing, Solver and Post-Processing. In pre-modelling, the dimension and specification of the model were determined. The minimum diameter of a tube was determined for a given pressure drop, length of tube, viscosity of fluid and volumetric flow rate of the fluid. In modelling, the 3-dimensional tube model was created with the ANSYS™ Design Modeller (DM). After the solid model of the flow domain has been created, the next stage is to process the solid flow domain in the ANSYS™ Meshing. In this stage, the domain is divided into small cells. Meshing is an essential processing stage as it provides a surface representation for a complex geometry with a few basic geometry primitives. During the meshing stage, the physical domain of the fluid flow is split into pieces of smaller 3D domains. The smaller domains are termed as sub-domains composing of the elements of simplices such as tetrahedrons. The model that has been meshed will be brought into the processing stage for further analysis. The whole process in this stage is under ANSYS™ Advance CFD. The inlet, outlet and wall condition for the model are specified in detailed. Then, the result is obtained from the conditions specified, and illustrated in a comprehensible graphical manner. The processing stage comprises of pre-processing stage, solver stage and post-processing stage.

The flow chart of the simulation of the coronary sinus conduit is shown in Fig.2. It comprises of a step-by-step procedure that is needed to generate the simulation. Meanwhile, the parameters of the CFD simulation for the models are summarized in Table 1.

RESULTS AND DISCUSSION

From all the results of the 3 models obtained, an overall analysis was been made. The graphs of the 3 models were combined to see their difference. For the pressure distribution comparison, the graph in Fig.3 was produced.

Table 1: Details of the models constructed

Parameter	Model 1	Model 2	Model 3
Fluid properties			
Equation	Power Law for Non-Newtonian Viscosity of Blood [5] $\eta = k\gamma^{n-1}e^{°T/T}$		
Power Law Index, n	0.4851	0.4851	0.4851
Consistency index, k ($kg.s^{n-2}/m$)	0.2073	0.2073	0.2073
Reference temperature (°T)	37	37	37
Maximum viscosity limit, μ_{max} (kg/m.s)	0.00125	0.00125	0.00125
Minimum viscosity limit, μ_{min} (kg/m.s)	0.003	0.003	0.003
Reference pressure (atm)	1	1	1
Fluid temperature (T)	37	37	37
Turbulance	Laminar	Laminar	Laminar
Density ($kg.m^{-3}$)	1050	1050	1050
Model properties			
Minor diameter, D_n (mm)	3	4	5
Major diameter, D_m (mm)	15	15	15
Tube length, l (cm)	11	11	11
Bending degree, α (°)	30	30	30
Part number, N	10	10	10
Throat diameter, D_T (mm)	1.13	1.15	1.17
Throat length, l_T (cm)	4	4	4
Mesh properties			
Total number of nodes	77704	77926	78297
Total number of tetrahedra	415245	416515	418579
Total number of elements	415245	416515	418579
Maximum spacing (mm)	0.5	0.5	0.5
Angle resolution (degree)	30	30	30
Minimum Edge Length (mm)	0.01	0.01	0.01
Maximum Edge Length (mm)	0.8	0.8	0.8
Simulation properties			
Inlet			
Mass flow rate ($kg.s^{-1}$)	-	-	-
Pressure (mmHg)	80	80	80
Outlet			
Mass flow rate ($kg.s^{-1}$)	0.004375	0.004375	0.004375
Pressure (mmHg)	-	-	-
Wall	Free Slip	Free Slip	Free Slip
No. Iteration	100	100	100

Through analytical analysis, Model 1 seems to be smoother than the other models. This proves that Model 1 experiences a steady decrease in pressure along the throat section. In addition, Model 1 gives an outlet pressure of 15 mmHg, which is most in accordance to the specification. Models 2 and Model 3 show a steep drop at the conduit length between 0.095 m to 0.085 m, which is not favorable due to the sudden contraction at the throat entrance zone.

Hence, details of model 1 will be discussed further. Based on the information illustrated in Fig.4, it is noticed that the entrance fluid pressure is about 80 mmHg. The pressure of fluid then decreases along the throat and the maximum reduction of the pressure can be observed to

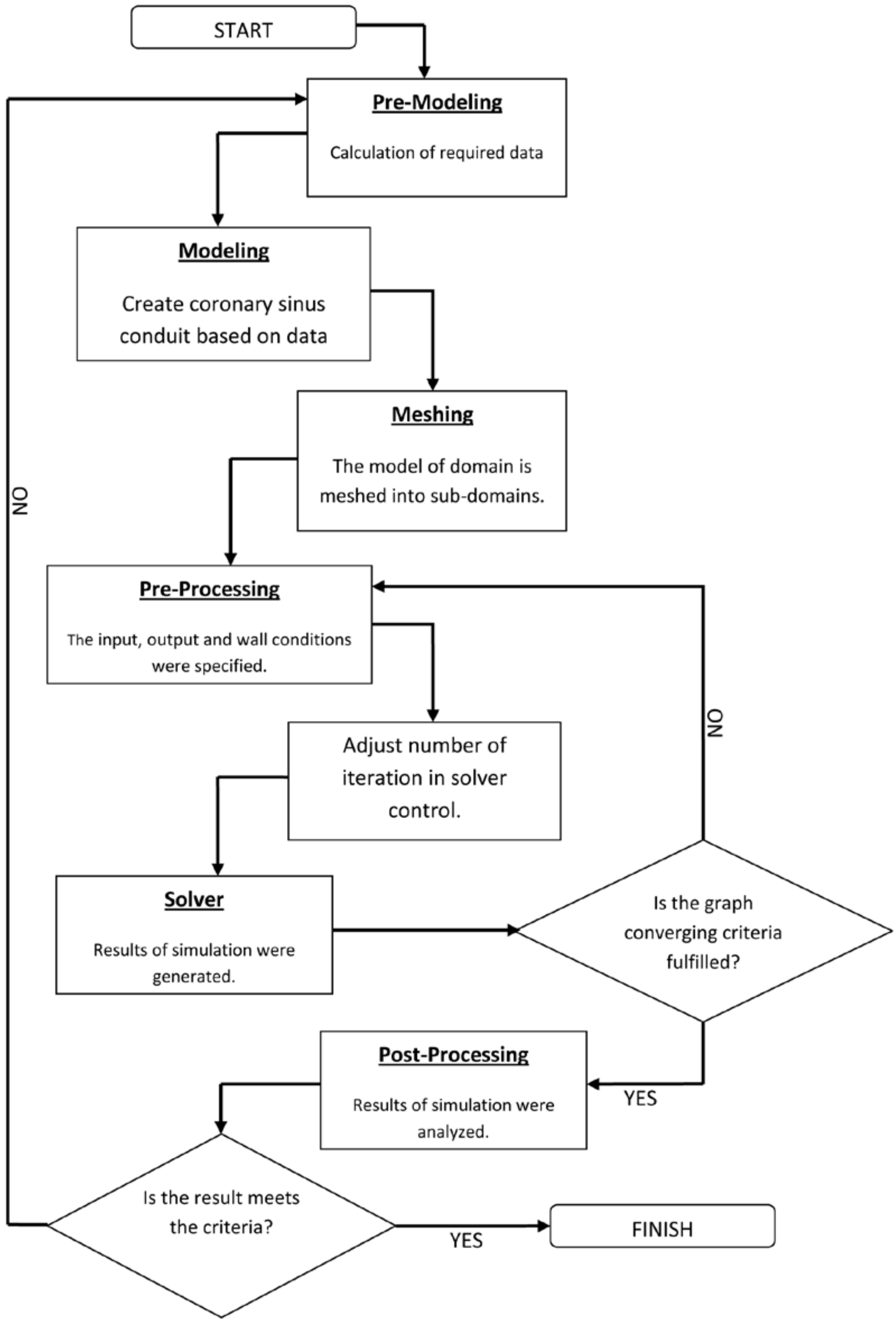

Fig.2: A flow chart of the simulation procedures using ANSYS™ CFX

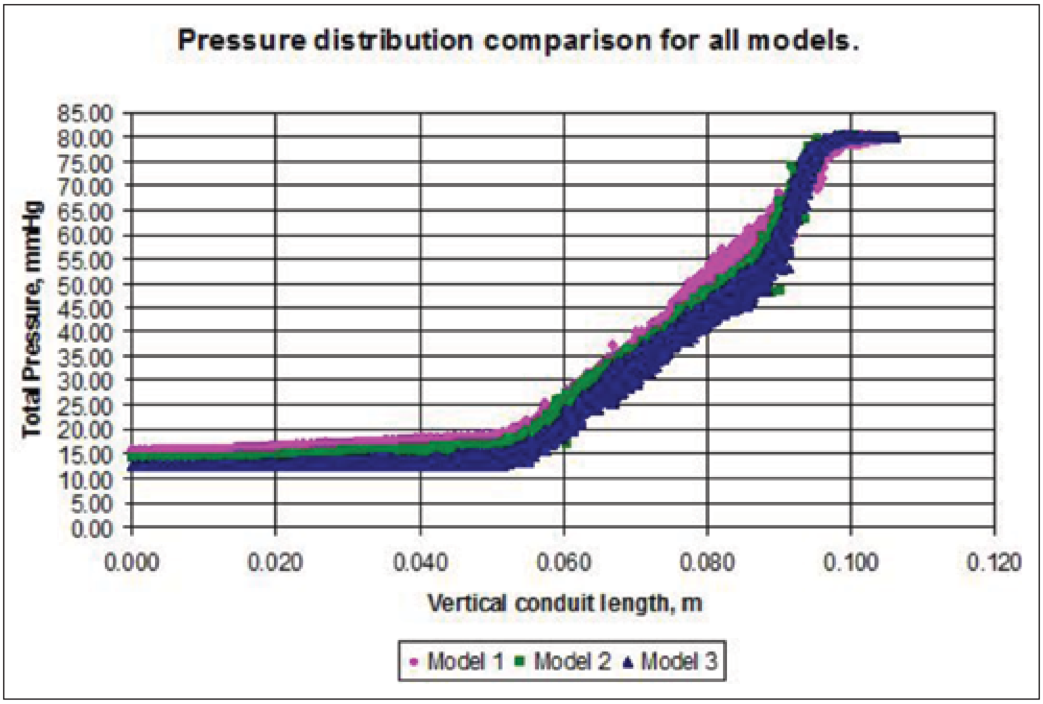

Fig.3: Pressure distribution comparison for all the models

happen at the inlet section of the throat. Towards the outlet of the throat, the pressure of the fluid increases back to a pressure of about 15 mmHg. This is due to the venturi effect, but it is to note that the conduit is not straight and the diameters of the inlet and outlet of the conduit are not uniform. A throat diameter of 1.13 mm can assure that it will not block the flow of blood. The maximum size of a red blood cell is about 8μm; by doing a simple calculation, a diameter of 1.13 mm can actually allow about 141 red blood cells to flow parallel in the throat section.

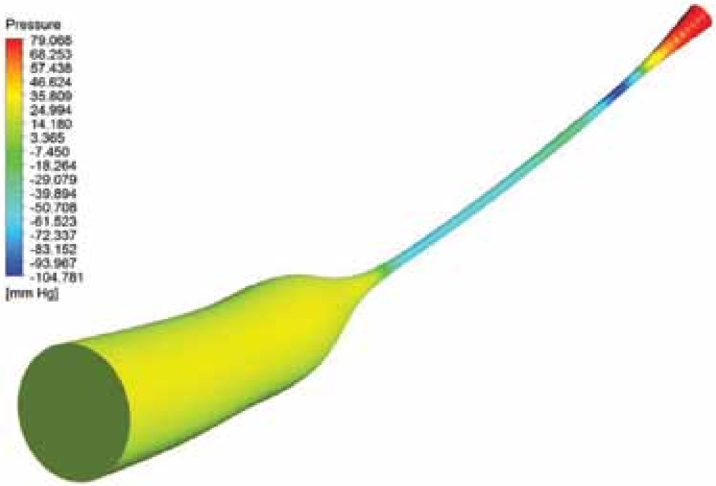

Fig.4: A contour plot on the external of conduit in Model 1

CONCLUSIONS

As a conclusion, Model 1 was found to be the best model. This is due to the characteristic of its stable pressure reduction along the throat and low peak velocity. In addition, Model 1 also provides a more uniform outlet pressure of 15 mmHg, which is in accordance with the required specification.

The objective of the current study was to investigate the pressure distribution of coronary sinus from the inlet pressure of 80 mmHg to the outlet pressure of 15 mmHg. With the utilization concept of the throat in the design, all the models successfully reduce the pressure from 80 mmHg at the inlet to merely 15 mmHg at the outlet. This pressure reduction effective zone is mainly on the throat section. It is crucial to note that this section was specifically designed to undergone venturi effect on the fluid flow passing through it. However, Model 1 with a 3mm inlet diameter and a 1.13 mm throat diameter was chosen as the best model because of the uniformity of the pressure reduced along the throat section.

Generally, the models provide a more flexible solution for blood flow in coronary sinus conduit. This is because in reality, the veins will undergo certain degree of contraction to push the fluid to move forward. In order to mimic this particular effect of contraction, the throat section was incorporated into the conduit.

ACKNOWLEDGEMENTS

First and foremost, the authors would like to express their deepest gratitude, appreciation and support to all the research team members, and also to En. Ismail A. Hamid, FRCS, for his advice and knowledge related to this project. This research filed for its patent by Innovation & Commercialisation Centre of Universiti Putra Malaysia on 16 November 2009, with the application number, PI20097028.

REFERENCES

Cengel, Y. A., & Cimbala J. M. (2007). *Fluid Mechanics: Fundamentals and Applications.* New York: McGraw-Hill Companies, Inc.

Lee, W. (2006). *Biofluid Mechanics in Cardiovascular Systems*. United States of America: McGraw-Hill Companies, Inc.

Petkova, S., Hossain, A., Naser, J., & Palombo, E. (2003). *CFD Modeling Of Blood in Portal Vein Hypertension with And without Thrombosis*. 3rd International Conference on CFD in the Minerals and Process Industries. Australia.

Syoten, O. (1980). Cardiovascular Hemorheology. Cambridge University Press. United Kingdom.World Health Organization - The world health report, 2007. Retrieved on January 27, 2010 from *www.who.int/whr/2007/whr07_en.pdf*.